

High Energy Boundary Conditions for a Cartesian Mesh Euler Solver

Shishir Pandya*

NASA Ames Research Center, Moffett Field, CA 94035

Scott Murman*

ELORET, Moffett Field, CA 94035

Michael Aftosmis[#]

NASA Ames Research Center, Moffett Field, CA 94035

1 Introduction

Inlets and exhaust nozzles are common place in the world of flight. Yet, many aerodynamic simulation packages do not provide a method of modelling such high energy boundaries in the flow field. For the purposes of aerodynamic simulation, inlets and exhausts are often faired over and it is assumed that the flow differences resulting from this assumption are minimal. While this is an adequate assumption for the prediction of lift, the lack of a plume behind the aircraft creates an evacuated base region thus effecting both drag and pitching moment values. In addition, the flow in the base region is often mis-predicted resulting in incorrect base drag.

In order to accurately predict these quantities, a method for specifying inlet and exhaust conditions needs to be available in aerodynamic simulation packages. A method for a first approximation of a plume without accounting for chemical reactions is added to the Cartesian mesh based aerodynamic simulation package CART3D^[1]. The method consists of 3 steps. In the first step, a components approach where each triangle is assigned a component number is used. Here, a method for marking the inlet or exhaust plane triangles as separate components is discussed. In step two, the flow solver is modified to accept a reference state for the components marked inlet or exhaust. In the third step, the flow solver uses these separated components and the reference state to compute the correct flow condition at that triangle.

The present method is implemented in the CART3D package^[1-4] which consists of a set of tools for generating a Cartesian volume mesh from a set of component triangulations. The Euler equations are solved on the resulting unstructured Cartesian mesh.

The present methods is implemented in this package and its usefulness is demonstrated with two validation cases. A generic missile body is also presented to show the usefulness of the method on a real world geometry.

[‡] Research Scientist, Member AIAA

* Senior Research Scientist

[#] Research Scientist, Senior Member AIAA

2 Technical Approach

Two tasks need to be accomplished in order to implement such a boundary condition. First, the set of triangles making up a inlet or nozzle need to be marked appropriately so that the flow solver can easily distinguish them from other triangles. Second, the flow solver must treat these triangles in a manner appropriate to the conditions specified by the user

2.1 Marking the triangulation

CART3D relies on a component wise approach to compose triangulated surfaces. In this approach, each component of the geometry is separately triangulated. All triangulated parts are then combined by intersecting the component triangulations. At the end of the intersection process, a single, closed, triangulated surface is obtained on which each triangle's origin can be identified by its component number.

A simple and effective strategy for identifying an inlet plane or an exhaust nozzle in this context is to assign a separate component number to the triangles that belong to an inlet or exhaust plane. However, inlet and/or exhaust planes are usually not modelled as separate components during the CAD process and often the geometry is an old triangulation that does not have its components identified. For these cases, a tool to extract the inlet and exhaust planes as components is used. Presently, the tool allows the user to specify either a bounding box(a rectangular cube) or a sphere to mark an inlet or exhaust region. Any triangle within these regions is marked by the tool according to the user specification.

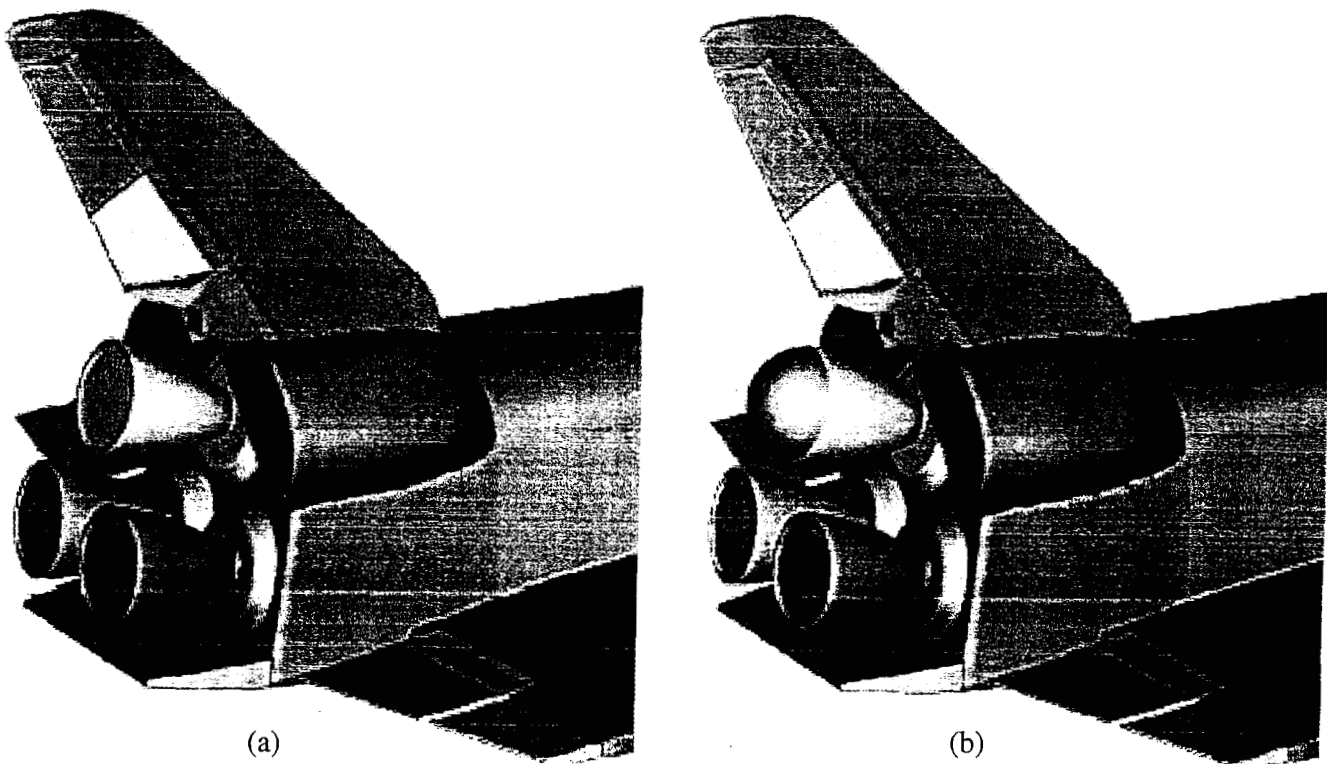


Figure 1: (a)The fared over exhaust nozzles of the space shuttle orbiter (b) A spherical cutter is placed over the area to be marked as exhaust

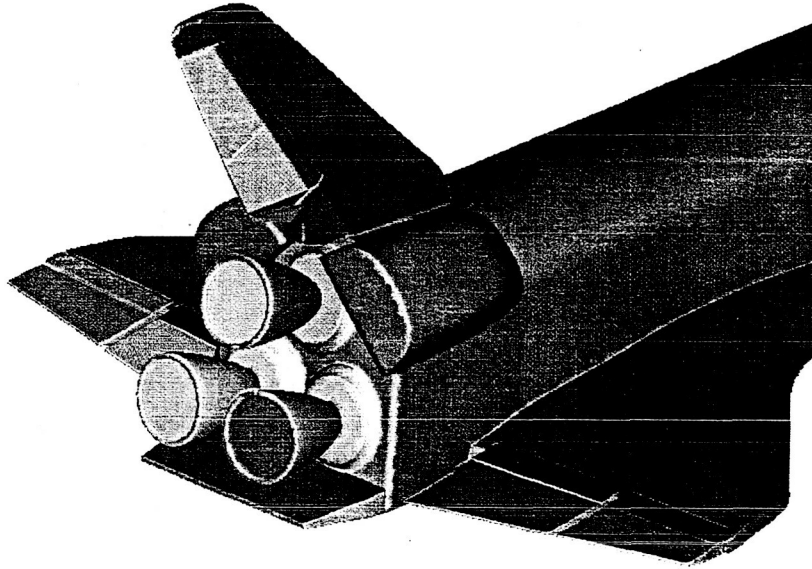


Figure 2: The exhaust planes of the three nozzles have been extracted as shown by the distinct colors of the components.

Figure 1 (a) shows a legacy geometry where at the back end of a shuttle orbiter the exhaust planes for the 3 Space Shuttle Main Engine nozzles need to be extracted as separate components. The user specifies a sphere as shown in Fig. 1 (b) to extract the exhaust plane of the top nozzle as a separate component. Figure 2 shows the resulting component marking. Each component is shown in a distinct color.

2.2 Flow solver algorithm

The boundary in the CART3D package is described by the surface triangulation. When the Cartesian mesh is generated, a set of cut cells is computed by intersecting the triangulation with the Cartesian cells forming a set of cut cells around the surface of the geometry. These cut cells are arbitrary polygons in 2D and polyhedra in 3D.

At an inlet or an exhaust plane, the user specified reference state is the flow condition at the boundary. As shown in Fig. 3 the flow condition at the boundary is denoted U_L . The flow condition in the cut-cell next to the boundary is reconstructed from the flow variables in the local neighborhood and is denoted U_R . A Riemann problem is then solved to compute the flux across that piece of the cut cell.

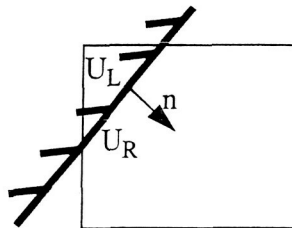


Figure 3: A typical cut cell. The Cartesian cell is cut by a boundary forming a cut cell.

The result is that for supersonic flow the boundary reference state specified by the user becomes the state at an exhaust plane. At a supersonic inlet, the result of the Riemann solver is to simply suck in whatever fluid is seen by the inlet plane. For subsonic flow, the Riemann solver computes an appropriate boundary value based on the characteristics of the flow and the specified boundary state.

An additional complication is that when the Riemann problem is solved in a non Cartesian-aligned plane, the velocity sent to the Riemann solver must be rotated into the coordinate system aligned with the normal to that boundary face. Once the Riemann problem is solved, the resulting flux must be rotated back to the original frame of reference and then added to the appropriate flux.

3 Results

Three cases are presented to validate the method. First, a simple box is used as the geometry to keep the geometric complexity from interfering with validation. This case has both an inlet and an exit and an exact solution exists allowing the verification of the angle of the shock (Fig. 4). The second case is that of an under expanded free jet. The solution is compared to experimental and computational values from Ref. 5. The third and final case is an application to a real configuration. The configuration of choice is a generic missile geometry.

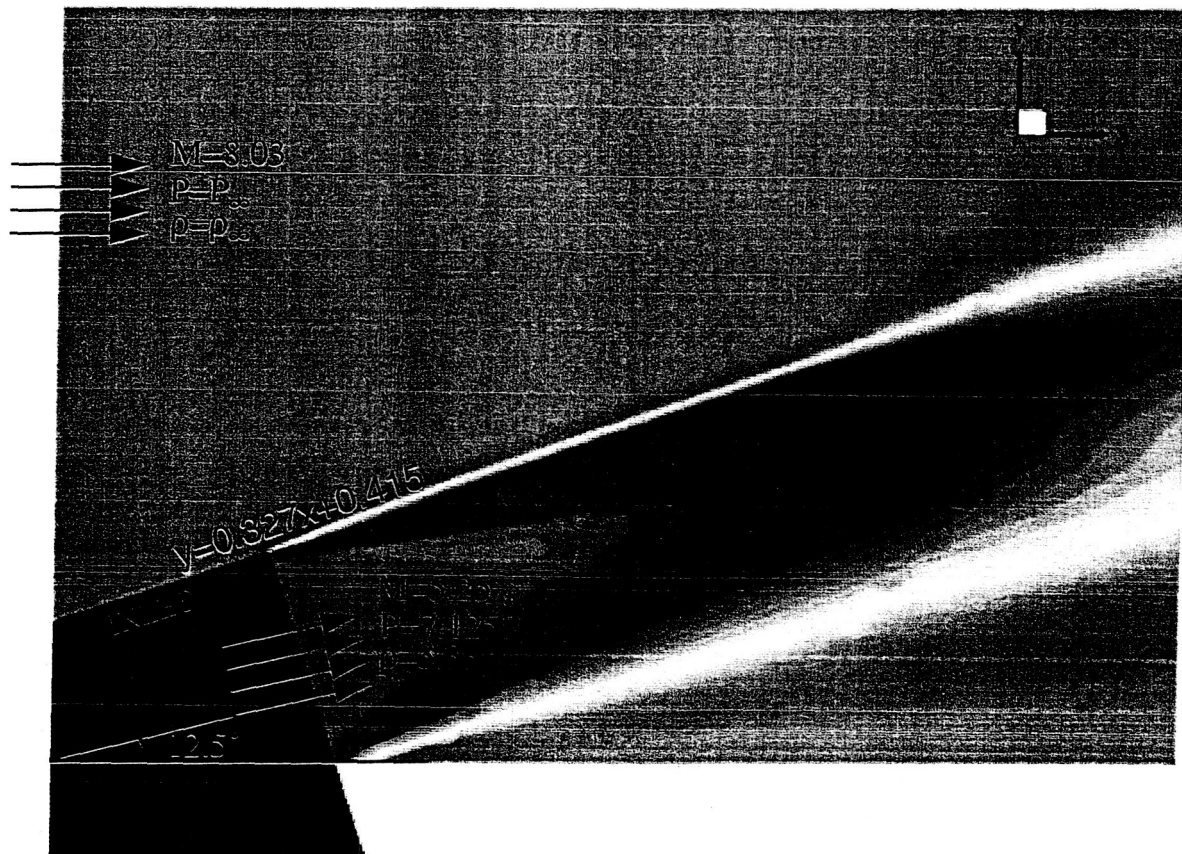


Figure 4: The solution over a box with exhaust conditions so that the resulting shock lines up with the boundary of the box. The conditions of the flow are shown on the figure.

3.1 Exact shock solution

For supersonic flow where the flow field receives fluid from 2 separate sources at two different conditions, a shock can be predicted analytically at the interface. Figure 4 shows such a case where one source of fluid is at $M=8.03$ at zero angle of attack and atmospheric conditions. A second source is admitting fluid into the flow field at a slower speed of $M=5.25$, but with higher pressure and density and at an angle of 12.5° to the horizontal. The exact solution to this flow field is a resultant linear shock at the interface which lies at an angle of 18.11° along the line described by $y=0.327x + 0.415$.

In Fig. 4 the box (shown in blue) is a triangulated geometry used to specify the second flow condition. The top surface of the box (marked INLET in Fig. 4) is marked as an inlet which at supersonic conditions acts like a simple extrapolation boundary "sucking in" all flow coming to it. The box is placed at an angle of 18.11° such that the shock should align with this boundary precisely. This however, complicates the exhaust condition on the right side of the box which must now release fluid at an angle to the physical boundary.

The solution is shown as color contours of pressure. Of note is the shock which recovers the predicted angle.

3.2 Under expanded Jet

A second test case for validation is that of an under expanded free jet. The results for this test case will be compared to an experiment conducted at Univ. of Virginia^[5]. This under expanded jet can be simulated using the present method of specifying high energy boundary conditions where the pressure at the exhaust plane is specified to be 20 times larger than the free stream pressure. A result of the present computational simulation is shown in Fig. 5. A detailed comparison with Ref. 5 will be presented in the paper.

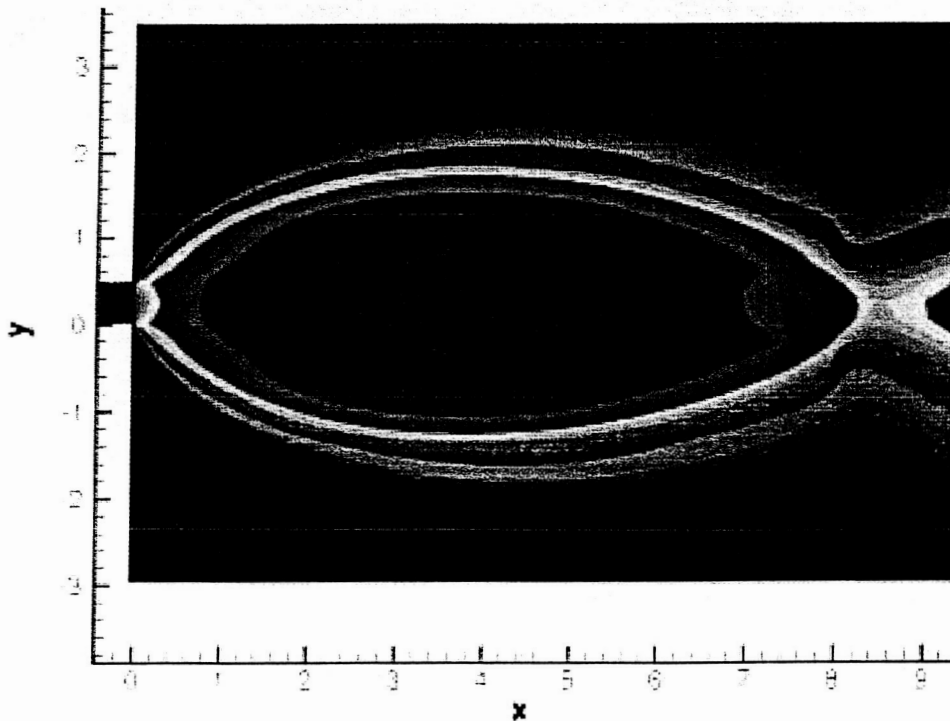


Figure 5: Under expanded jet with exhaust pressure ratio of 20 with respect to back pressure (free stream).

3.3 FM3

The more practical geometry of a generic missile body is chosen as the test case and a nozzle plane is marked as exhaust at the back of the missile. The missile is simulated at a speed of Mach 1.5 with the exhaust plane set to release fluid at Mach 2.5 and $P=10P_\infty$. An inviscid simulation of this missile has been performed with the CART3D package^[6] and a viscous simulation was performed with the OVERFLOW package^[7]. Both of these prior simulations were performed without a plume. However, these simulations were done with the missile in constant roll. We will not model the roll for the purpose of demonstrating the exhaust boundary conditions.

The resulting velocity field is shown in Fig. 6. The bow shock in front of the missile as well as the shocks from the canard are visible as expected for this missile^[6]. An additional feature of this flow is the nozzle plume resulting from the under expanded jet. Because this plume expands into the flow field, it causes an additional set of shocks ahead of the plume and near the tail of the missile body.

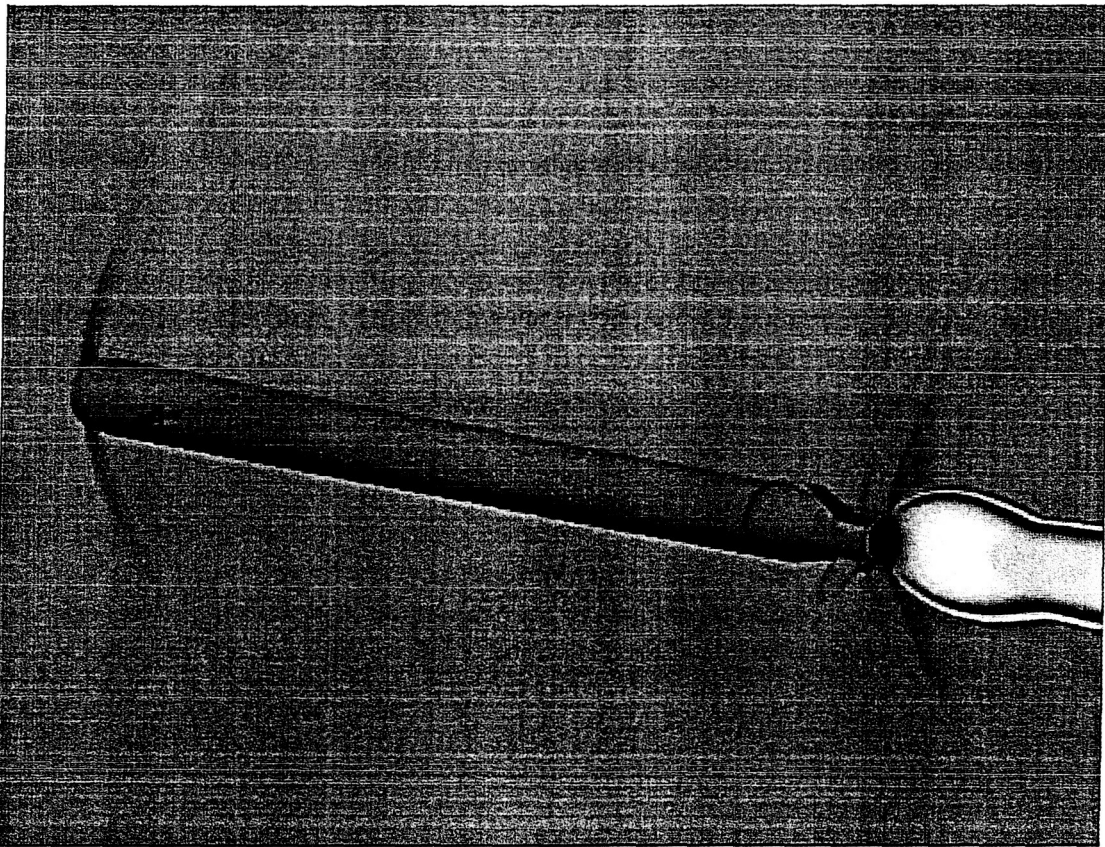


Figure 6: Axial velocity contours over a generic missile body with canard and fins. The plume conditions are set to Mach 2.5 and a pressure ratio of 10 with respect to the free stream.

A plot of density is shown to clarify the structures of the shocks in Fig. 7. The plume replaces the base region of the missile, and because the plume is an under expanded jet, it causes the flow to move around the plume. The result of this is that the larger the plume, the stronger the shock. At angle of attack, the shock takes on different positions and strengths on different sides of the body

thus changing the resulting pitching moment. A plume of this kind is very helpful, therefore, in obtaining realistic pitching moments for missile configurations.[†]

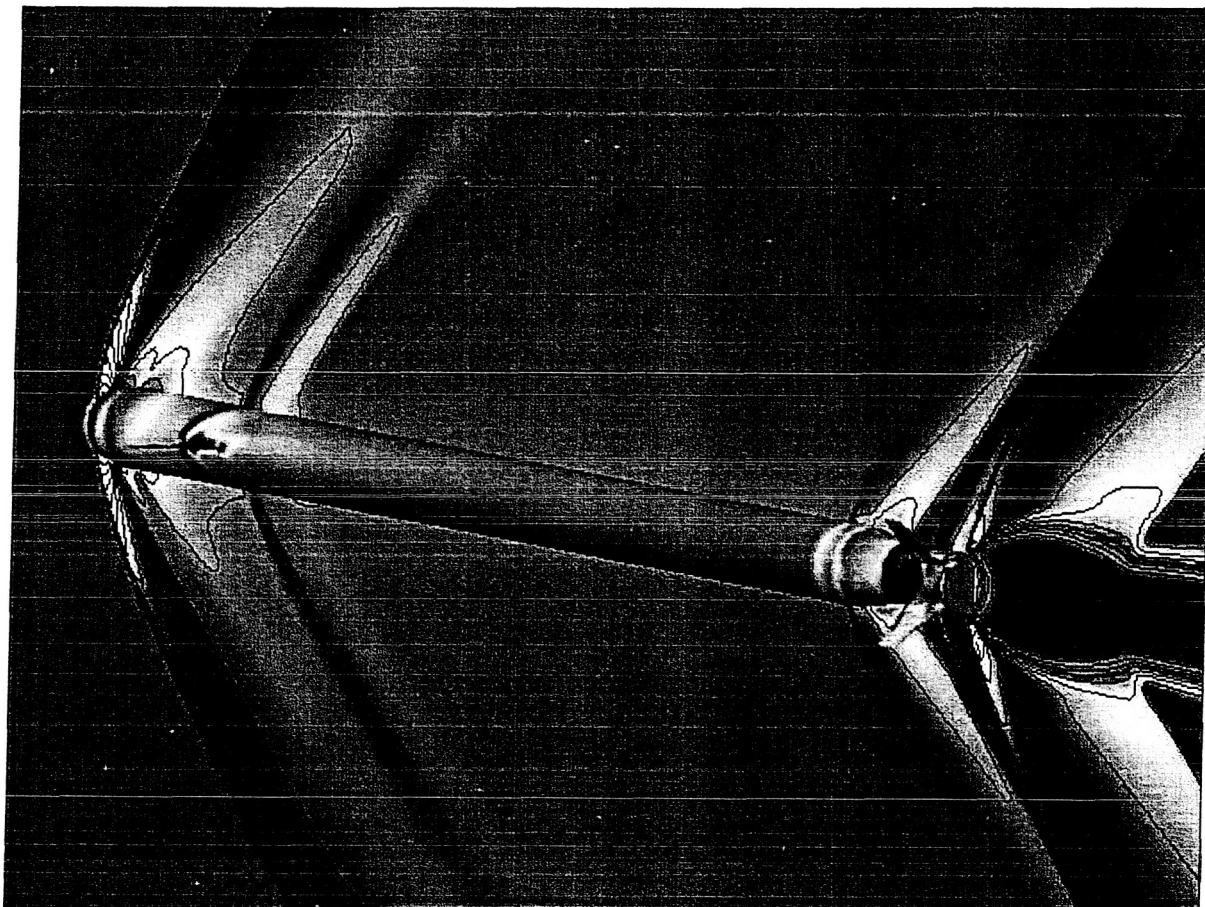


Figure 7: Density contours over the generic missile body. The shock structure on the rear of the body resulting from the presence of the plume can be seen.

4 Summary

A method for the first approximation of an inlet or exhaust has been presented. The method is validated with two test cases. The test cases confirm that an appropriate condition set at the inlet and exhaust behaves as expected by theory or experiment. A further case of a generic missile body is used to demonstrate the usefulness of the method for real world geometries.

The paper will include a more detailed validation and further data evaluation to show the usefulness of the method in prediction of forces and moments on bodies such as the missile and space shuttle.

5 References

- [1] Aftosmis, M. J., Berger, M. J., Melton, J. E., "Robust and efficient Cartesian mesh generation for component-based geometry" *AIAA Paper 97-0196*, Jan. 1997.
- [2] Aftosmis, M. J., Berger, M. J., and Adomovicius, G., "A parallel multilevel method for adaptively refined Cartesian grids with embedded boundaries" *AIAA Paper 2000-0808*, Jan. 2000.

- [3] Aftosmis, M. J., Berger, M. J., and Adomovicius, G. "Parallel multigrid on Cartesian meshes with complex geometry" *Proc. of the 8th Intl. Conf. on Parallel CFD*. Trondheim Norway. (To appear Elsevier Press). Jun. 2000.
- [4] Melton J. E., Berger, M. J., and Aftosmis, M. J., "3D applications of a Cartesian grid Euler method," *AIAA Paper 95-0853-CP*, Jul. 1993.
- [5] J. Mc Daniel, c. Glass, D. Staak, C. Miller, "Experimental and Computational Comparison of an underexpanded Jet Flowfield", *AIAA Paper 2002-0305*, Jan. 2002, Reno, NV.
- [6] Murman, S. M., Aftosmis, M. J., and Berger, M. J., "Numerical Simulation of Rolling Airframes Using a multi-level Cartesian Method" *AIAA Paper 2002-2798*. Jun. 2002.
- [7] Buning, P. G., Jespersen, D. C., Pulliam, T. H., Chan, W. M., Slotnick, J. P., Krist, S. E., and Renze, K. J., "OVERFLOW User's Manual" NASA.